

V94-24446

1993

NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

**MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA IN HUNTSVILLE**

**MICROMECHANICAL SIMULATION OF DAMAGE PROGRESSION IN CARBON
PHENOLIC COMPOSITES**

Prepared By:	Kerry T. Slattery, Ph.D.
Academic Rank:	Assistant Professor
Institution and Department:	Washington University in St. Louis Department of Civil Engineering
MSFC Colleagues:	Raymond G. Clinton, Ph.D. Roy M. Sullivan, Ph.D.
NASA/MSFC:	
Office:	Materials and Processes Laboratory
Division:	Nonmetallic Materials
Branch:	Ceramics and Coatings

INTRODUCTION

Carbon/phenolic composites are used extensively as ablative insulating materials in the nozzle region of solid rocket motors. The current solid rocket motor (RSRM) on the space shuttle is fabricated from woven rayon cloth which is carbonized and then impregnated with the phenolic resin. These plies are layed up in the desired configuration and cured to form the finished part. During firing, the surface of the carbon/phenolic insulation is exposed to 5000°F gases from the rocket exhaust. The resin pyrolyzes and the material chars to a depth which progresses with time. The rate of charring and erosion are generally predictable, and the insulation depth is designed to allow adequate safety margins over the firing time of the motor. However, anomalies in the properties and response of the carbon/phenolic materials can lead to severe material damage which may decrease safety margins to unacceptable levels. Three macro damage modes which have been observed in fired nozzles are: ply lift, "wedge out", and pocketing erosion. Ply lift occurs in materials with plies oriented nearly parallel to the surface. The damage occurs in a region below the charred material where material temperatures are relatively low — about 500°F. Wedge out occurs at the intersection of nozzle components whose plies are oriented at about 45°. The corner of the block of material breaks off along a ply interface. Pocketing erosion occurs in materials with plies oriented normal to the surface. Thermal expansion is restrained in two directions resulting in large tensile strains and material failure normal to the surface. When a large section of material is removed as a result of damage, the insulation thickness is reduced which may lead to failure of the nozzle due to excessive heating of critical components. If these damage events cannot be prevented with certainty, the designer must increase the thickness of the insulator thus adding to both weight and cost.

One of the difficulties in developing a full understanding of these macro damage mechanisms is that the loading environment and the material response to that environment are extremely complex. These types of damage are usually only observed in actual motor firings. Therefore, it is difficult and expensive to evaluate the reliability of new materials. Standard material tests which measure mechanical and thermal properties of test specimens can only provide a partial picture of how the material will respond in the service environment. The development of the ANALOG test procedure (2) which can combine high heating rates and mechanical loads on a specimen will improve the understanding of the interactive effects of the various loads on the system. But a mechanistic model of material response which can account for the heterogeneity of the material, the progression of various micromechanical damage mechanisms, and the interaction of mechanical and thermal stresses on the material is required to accurately correlate material tests with response to service environments. A model based on fundamental damage mechanisms which is calibrated and verified under a variety of loading conditions will provide a general tool for predicting the response of rocket nozzles. The development of a micromechanical simulation technique has been initiated and demonstrated to be effective for studying across-ply tensile failure of carbon/phenolic composites.

APPROACH

The finite element method is used to simulate the progression of micromechanical damage mechanisms in the carbon/phenolic material. Two damage mechanisms are considered: fiber/matrix interface debonding and matrix cracking. The failure process in across-ply tension appears to initiate at the fiber/matrix interface and progress to adjacent fibers. A crack eventually reaches the interface between two plies and propagates along that interface resulting in specimen rupture. Fiber breakage is observed where yarns are severely kinked, but this damage mode is assumed to occur after the development of a critical flaw and is not currently accounted for in the model.

A two-dimensional finite element model is created to simulate the failure of a section of the composite. A typical model consists of one yarn end along with parts of the surrounding in-plane yarns. A sketch of a typical model is shown in Fig. 1. The model consists of three types of finite elements: out-of-plane fiber (OPF), in-plane fiber (IPF), and matrix (MAT). The elements are square with four-nodes and eight-degrees-of-freedom. The OPF element represents a fiber end surrounded by a small amount of matrix. The IPF element has the same dimensions and represents a composite oriented at the yarn angle at the element location. The MAT element is pure matrix and is placed in resin-rich areas. The OPF, IPF, and MAT are "superelements" whose properties are determined from detailed finite element analyses of the constituent materials. Stiffness, thermal expansion, and crack-tip displacement properties are tabulated for many possible damage states for each superelement type. For example, damage in the OPF element is characterized by the location and length of debonds along the fiber/matrix interface. Finite element models are generated and analyzed for approximately 1000 different debond configurations. The results are stored and used to determine superelement properties in the simulation based on the initial interface flaws and the progression of those flaws. This method allows efficient simulation of micromechanical damage progression on models of significant sections of composite.

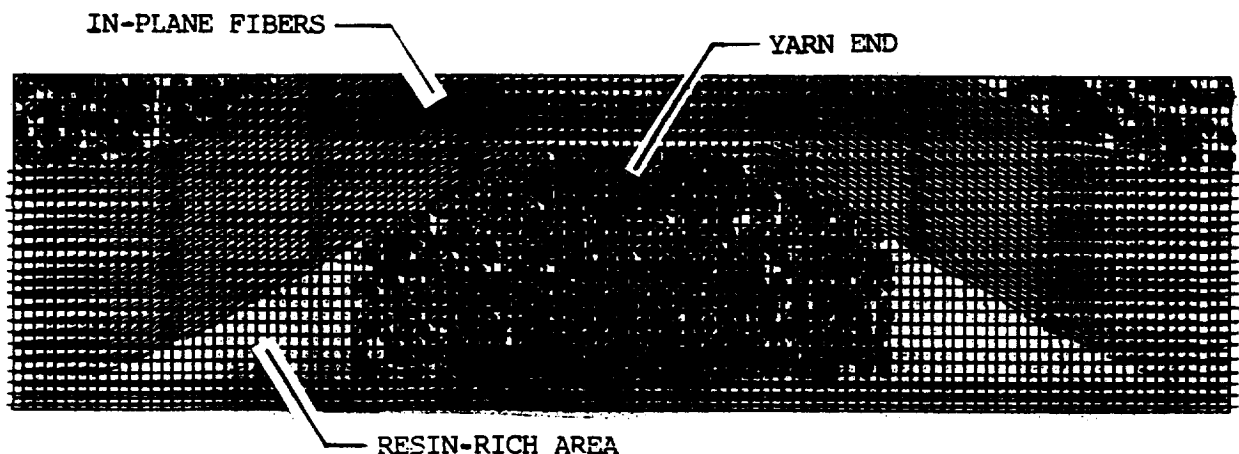


Figure 1. Micromechanical Simulation Model of Woven Carbon/Phenolic Composite

The damage growth model is based upon fracture mechanics principles. A simple model for initial flaws is assumed at the beginning of the simulation. All initial flaws are on the fiber/matrix interface. In the detailed finite element model of the OPF, there are 32 nodes on the interface. An interface flaw is modeled by "disconnecting" the fiber from the matrix at a node. A large debond is formed when several adjacent nodes are disconnected. Flaw distribution schemes are usually random. The simulation method allows the flexibility to investigate a variety of flaw configurations. The two used in this work were placing a fixed length debond (e.g. 45 degrees) on some percentage of randomly selected fibers and specifying a percentage of disconnected nodes on the fiber/matrix interface. Flaw growth is determined using the crack closure method. This method has been used to study failure modes in metal matrix composites (1). Each existing flaw in a superelement is analyzed in several possible propagated states given the current nodal displacement. Tabulated data on crack tip displacements are used to determine the distance between the nodes at the current crack tip and the displacement caused by a unit force at those nodes. The amount of work required to close the crack to its current state from the assumed, propagated state is calculated and compared with the amount of energy required to create the new surface. The crack propagates if the work exceeds the surface energy. The model is idealized since the fibers, which are modeled as circular, actually have irregular shapes and since the quality of the bond between the fiber and matrix also varies around and along the fiber; however, the interface model should provide sufficient flexibility to adequately match the response of the interface by varying the surface energy and the flaw distribution.

A material configuration is selected based on photomicrographs of the composite. A simple mesh generation subroutine is written to define the distribution of the three types of elements and the direction of the IPF elements in the finite element simulation. The nodes on the bottom of the model are fixed, and a uniform tensile stress is applied to the opposite face. The stress level is increased in small increments, and the model is analyzed. After each load step, the properties of each element are updated based upon the crack propagation models. The simulation continues until a maximum stress is reached and severe damage occurs in the model.

RESULTS

Figure 2 shows the progression of damage in a simple section of out-plane fibers with some pure matrix elements. Initial flaws on the fiber/matrix interface are represented by thick lines in Fig 2a. These initial flaws begin to progress at about 0.07% strain as shown in Fig. 2b. The interface flaws propagate to adjacent fibers and eventually coalesce to form a critical flaw which leads to specimen rupture as shown in Fig. 2c. The technique was also applied to a more complex model such as that shown in Fig. 1. The results of many simulations using a range of values for various parameters demonstrated that the response of carbon/phenolic materials can be simulated effectively using this technique.

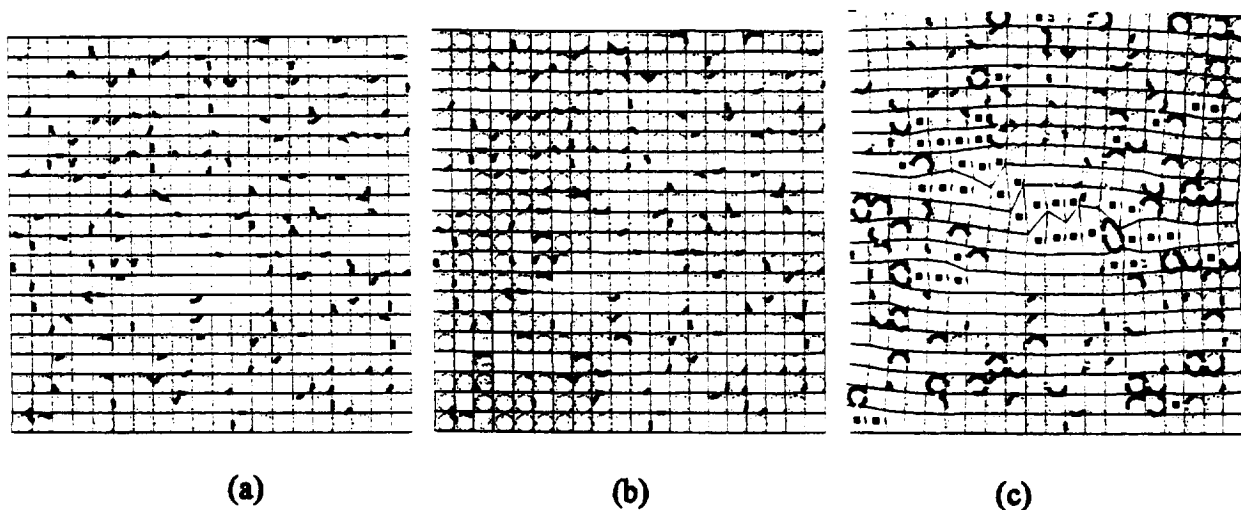


Figure 2. Damage Progression in Composite Loaded in Transverse Tension

CONCLUSIONS

A technique to perform micromechanical simulations of damage progression in carbon/phenolic composites has been developed. The technique is effective at modeling across-ply tensile response although additional calibration and verification based on damage in tested specimens must be performed to refine the estimates of critical parameters. Thermal loads can also be applied in the simulation, and preliminary results demonstrate that cracking during post-cure cooldown can be predicted using this technique. Given values for the three principle model parameters: fiber/matrix interface surface energy, interface flaw distribution, and matrix surface energy, along with standard material properties for the constituent materials, any loading condition can be easily simulated. Of course, some of these properties cannot be measured directly, so the simulation technique can aid in determining these values by performing simulations of the material response under a variety of loads and finding the optimum values for the parameters which yield the best results for most conditions. This method can also be extended to three-dimensions if extensive computer resources are available, but two-dimensional simulations can provide substantial new insights into the behavior of carbon/phenolic composites.

REFERENCES

1. Mital, S.K., Caruso, J.J., and Chamis, C.C., "Metal Matrix Composites Microfracture: Computational Simulation," *Computers & Structures*, Vol. 37, No. 2, February, 1990, pp. 141-150.
2. Poteat, R.M., Ohler, H.C., Koenig, J.R., Wendel, G.M., Crose, J.G., and Marx, D.A., "Nozzle Ablative Simulation Apparatus Development," *Proceeding of JANNAF Rocket Nozzle Technology Subcommittee Meeting*, December 1992.